IN-64

51

NASA Contractor Report

Addendum to the DYCAST User's Manual Describing the Curved, Warp Beam Finite Element

L. M. Kutt and A. B. Pifko

CONTRACT NAS1-18444
MAY 1991

NASA Contractor Report 4362

Addendum to the DYCAST User's Manual Describing the Curved, Warp Beam Finite Element

L. M. Kutt and A. B. Pifko
Grumman Corporate Research Center
Bethpage, New York

Prepared by Grumman Corporate Research Center
for Lockheed Missiles & Space Company, Inc., for
Langley Research Center
under Contract NAS1-18444

National Aeronautics and
Space Administration
Office of Management
Scientific and Technical
Information Division

1991

PREFACE

DYCAST is a finite element computer program developed at the Grumman Corporate Research Center (with partial support from the NASA Langley Research Center) for the nonlinear transient dynamic analysis of structures. As part of the Computational Structural Mechanics (CSM) research program of NASA, the Applied Mechanics Laboratory at the Grumman CRC is developing capabilities in its DYCAST software system for the analysis of aerospace structures made of composite materials. One result of this development effort is that a curved, linear, thin-walled beam element has been incorporated into the DYCAST library of finite elements. The formulation of this new element, called the WARP element, includes both shear deformations and nonuniform torsion (restrained warping). The FORTRAN software that is responsible for the calculation of the WARP beam element stiffness and mass matrices and related quantities and the calculation of the cross-sectional properties was made available by Professor Ahmed K. Noor of the University of Virginia, who is also the Director of the NASA Center for Computational Structures Technology, for incorporation into DYCAST.

This report describes the basic concepts, the input data, and several example problems essential to the DYCAST analyst for the successful use of the WARP element. It supplements the existing user's manual for DYCAST (Ref 1). The sample analyses in this report, along with many others, verify the excellent performance of the WARP element, both from the point of view of physical behavior and numerical stability.

This work was supported by NASA Contract NAS1-18444, the "Development of Finite Elements for Composite Structures and Inclusion into the DYCAST Computer Code," Task 8 of the Computational Structural Mechanics (CSM) research effort of the NASA Langley Research Center. The help and continued interest of Mr. Huey D. Carden of the NASA Langley Research Center is gratefully acknowledged.

CONTENTS

ILLUSTRATIONS

TABLES

1. OVERVIEW OF THE WARP BEAM ELEMENT

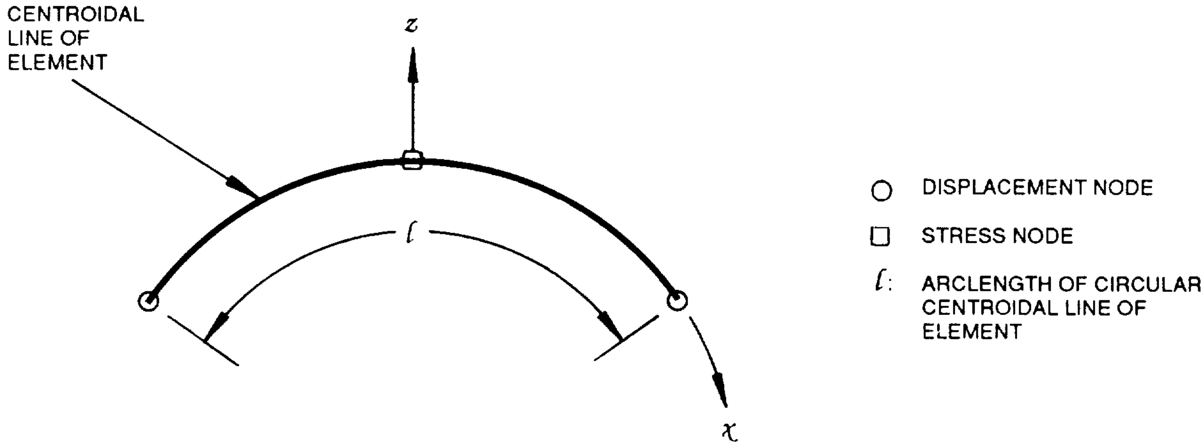
The WARP beam element of DYCAST is applicable to the finite element analysis of thin-walled structures, i.e., beams, columns, frames, stiffeners, etc. The analytical and finite element formulation is that of Noor et al. (Ref 2). The element takes into account the coupling of axial, flexural, and torsional behavior, with the effects of transverse shear deformation and rotary inertia included. The longitudinal axis of the beam element can be either curved or straight, and the cross section is assumed constant along this axis (i.e., prismatic) and open. In its present form, the material is assumed to be linear, elastic, isotropic, and homogeneous, and geometric nonlinearities are excluded from the formulation. Thus, linear static and linear free-vibration analyses can be performed with the present version of the WARP element, but nonlinear transient dynamic analyses cannot. Future developments will include material anisotropy, failure criteria, and geometric nonlinearities.

In the following sections, we explain basic concepts about the WARP beam element that are essential for its successful use.

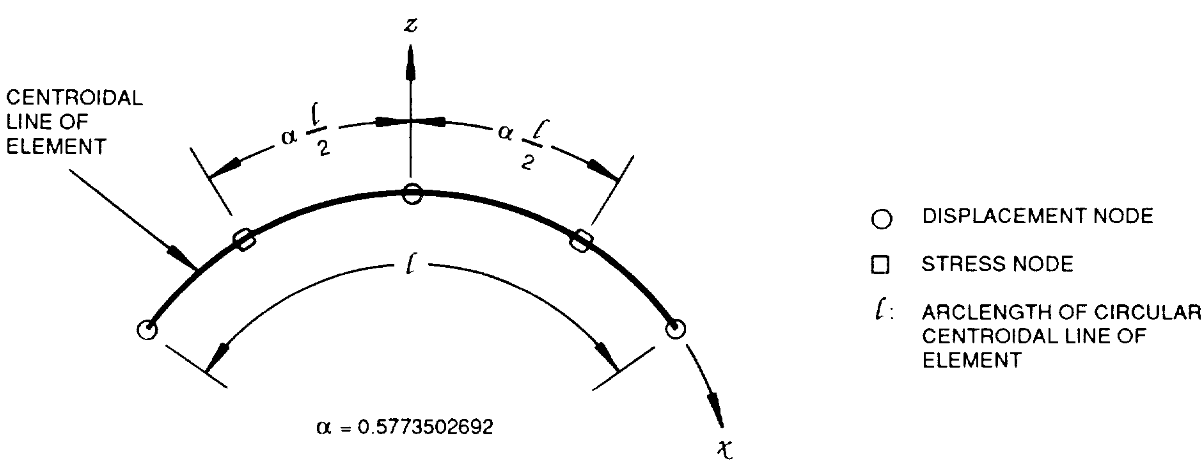
1.1. DISPLACEMENT & STRESS NODES

A perturbed Lagrangian - mixed variational principle forms the basis of the thin-walled WARP beam finite element. Thus, this beam element possesses both displacement nodes and stress nodes. There are seven degrees of freedom at every displacement node: the three translational displacements, the three angles of rotation, and the rate of twist θ. In DYCAST, the user can specify either two or three such displacement nodes per element. This is done via the element connectivity WARP cards discussed in section 2.1. In the case of two nodes, one node is located at each of the end points of the element. In the case of three nodes, an additional node is located at the midpoint along the length of the beam, i.e., the three displacement nodes are equally spaced.

Upon specifying the number of displacement nodes, the DYCAST program automatically assigns the number of stress nodes and their locations. Thus, in the case of two displacement nodes, there is one stress node located at the midpoint along the length of the beam, as depicted in Fig. 1a. In the case of three displacement nodes, there are two stress nodes, as depicted in Fig. 1b. Note from Fig. 1b that the



(a) TWO DISPLACEMENT NODES PER ELEMENT



(b) THREE DISPLACEMENT NODES PER ELEMENT

R90-3042-001

Fig. 1 Beam Element Nodes

x-coordinates of the stress nodes coincide with the Gaussian quadrature coordinates for the case of two integration points. This concept of situating the stress nodes at quadrature points rather than at the locations of the displacement nodes has been demonstrated by Noor et al. (Ref 3) to greatly increase the accuracy of computations.

The theory upon which this WARP beam element is based is characterized by seven stress resultants: the axial force, the two shear forces, the two bending moments, the torsional moment (or torque, or twisting moment), and the bimoment (or warping moment). These seven static variables are the stress degrees of freedom of stress nodes. It should be emphasized, however, that these variables, along with the Lagrange multiplier parameters, are statically condensed out before the assembly of the elements into the global equations of the structural system (Ref 2). Thus, in the finite element modeling of a structure, the user generates only displacement nodes, and beam element connectivity is defined only in terms of the displacement node numbers. The stress nodes remain "invisible" to the user, unless the user is interested in printing out the stress resultant variables. In other words, these stress nodes are internal nodes.

When the beam element has three displacement nodes and two stress nodes, the element shape functions for the generalized displacement fields are quadratic. The shape functions for the stress resultant fields and the Lagrange multiplier are linear. In this case, three-Gaussian-quadrature-point numerical integration is used to compute the element stiffness matrix and mass matrix.

Similarly, in the case of two displacement nodes and one stress node, the element generalized-displacement fields are interpolated linearly, the stress resultant fields and the Lagrange multiplier are constant throughout the beam element length, and two-point Gaussian integration is used.

The WARP beam element can have any arbitrary orientation in three-dimensional space that is characterized by some global Cartesian XYZ coordinate system. This is because the three translational displacements and the three rotational displacements at each displacement node, and the corresponding element stiffness and inertia coefficients, are transformed into the global Cartesian XYZ coordinate system. However, the seventh degree of freedom, i.e., the rate of twist θ, is <u>not</u> transformed. The rate of twist remains defined in the local coordinate system of the element. Care should therefore be exercised in connecting two or more WARP beam elements to a particular node in a given finite element model. For every beam element connecting to a common node, consider the vector that is tangential to the longitudinal centroidal x-axis of the beam element and is located at this common node. In order to use the WARP element properly, all such tangential vectors must be parallel, or closely parallel. Thus, unless coincident nodes and multipoint constraints are used, there can be no large "kinks" in an arch, and a beam and a column cannot in general intersect at right angles (or <u>any</u> nonzero "large" angle).

In the previous discussion, nonzero "angles of intersection" of WARP elements were addressed. Coincident nodes and multipoint constraints may also be necessary in situations such as:

1. Two parallel, straight WARP beam elements connecting to a common node, but having cross sections of different shape and/or size (see Ref. 4 for a discussion on connections);

2. A WARP beam element (with seven displacement degrees of freedom per node) connecting to a different type of finite element, such as a plate element with five displacement degrees of freedom per node.

1.2 COORDINATE SYSTEMS & DYCAST OUTPUT

The local, right-handed, orthogonal xyz coordinate system of the WARP beam finite element is curvilinear, with the curved x-coordinate axis coinciding with the longitudinal centroidal axis of the beam. (Refer to Fig. 1, 2, and 3.) The origin of the local element coordinate system, i.e., the point where x=y=z=0, is located at the midpoint of the curved centroidal line that runs between the two end points of the beam element. The curvature of the x-axis is in one direction only, i.e., the x-and z-axes lie in a plane, and the y-axis is perpendicular to this plane. Because of the centroidal nature of the x-axis, <u>all</u> of the nodal stress resultants (including the twisting moment and the bimoment) and <u>all</u> of the generalized strains (including the curvature changes, the twist κ_t, and the strain parameter ψ) refer to the <u>centroidal</u> axis, <u>not</u> the axis of shear centers. Stated differently, the (internal) stress nodes are located on the centroidal x-axis. In static analyses with the WARP beam element, DYCAST prints out the nodal stress and strain quantities in the local centroidal xyz element coordinate system. For specifics of the definitions and the sign conventions of these stress and strain variables, see Ref 2.

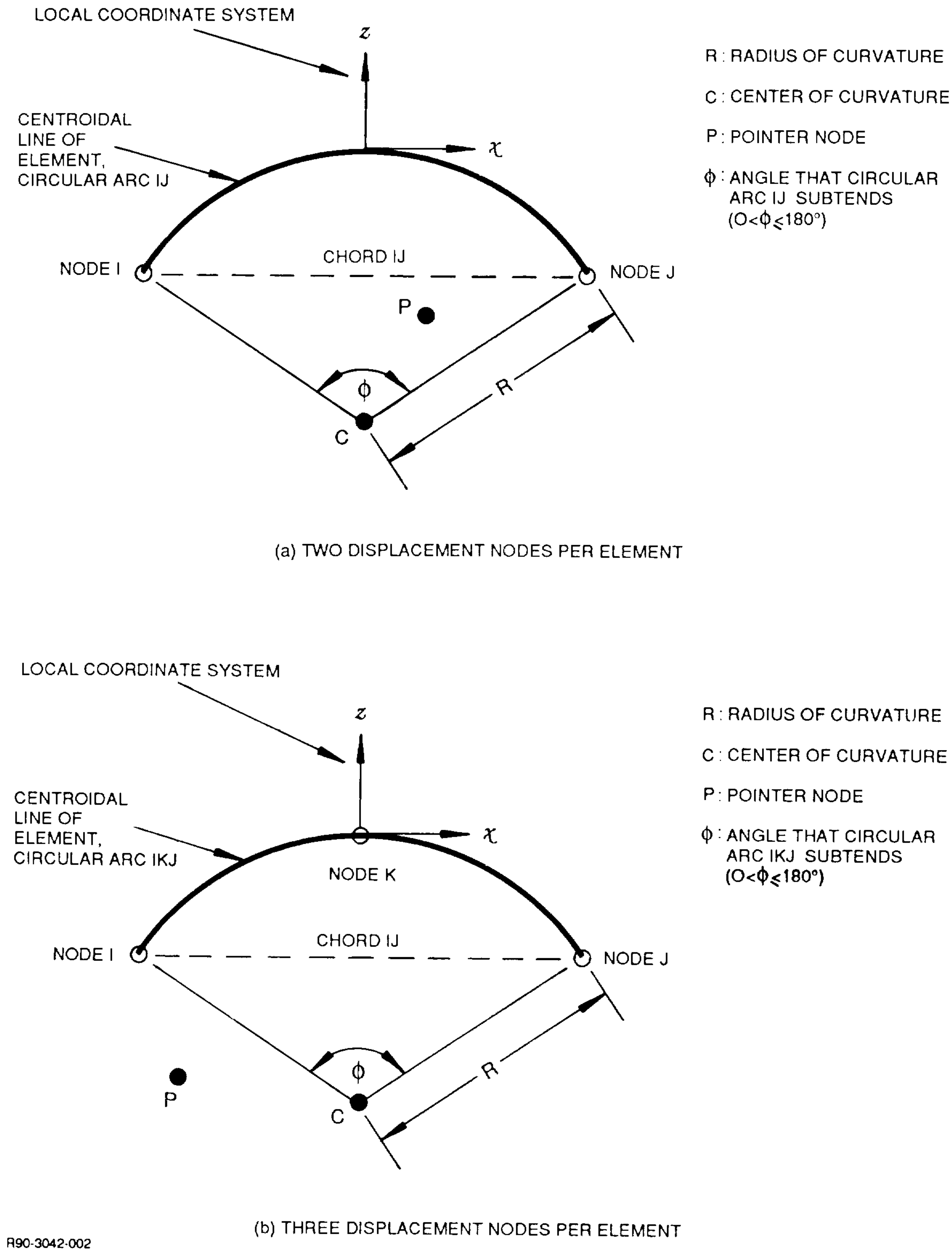
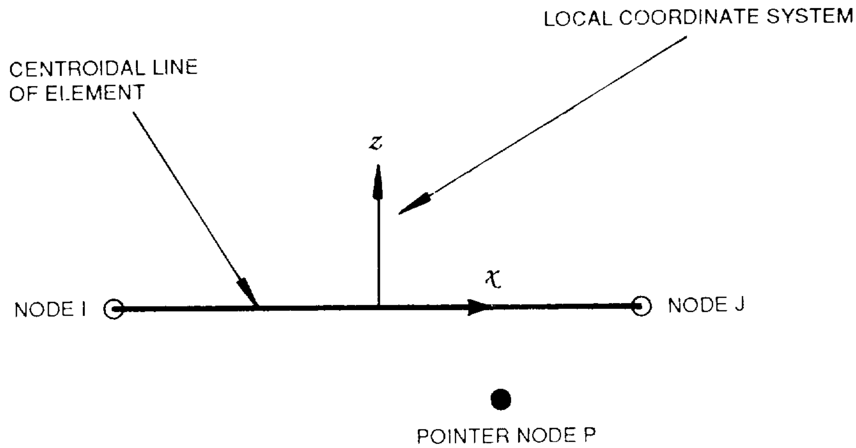
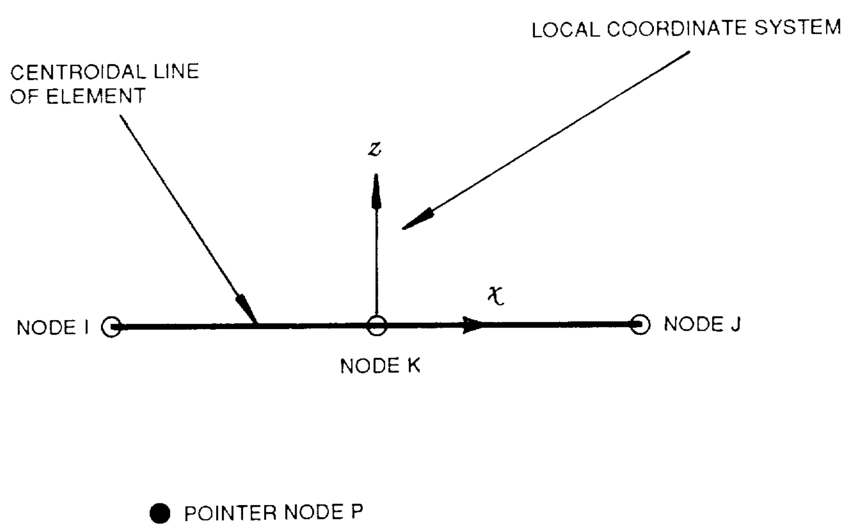


Fig. 2 Beam Element Curvature



(a) TWO DISPLACEMENT NODES PER ELEMENT



(b) THREE DISPLACEMENT NODES PER ELEMENT

R90-3042-003

Fig. 3 Straight Beam Element

Similarly, the three translational degrees of freedom of a displacement node are the deflections of the centroids, and the three rotational degrees of freedom are the angles of rotation of the beam cross section about three orthogonal axes passing through the centroid. Stated differently, the displacement nodes are located on the centroidal x-axis of the beam. Unlike the situation with the stress nodes, however, DYCAST prints out these six nodal displacement degrees of freedom in the global Cartesian XYZ coordinate system. Therefore, the three rotational degrees of freedom that are printed by DYCAST are the angles of rotation of the beam's cross section about three coordinate axes that:

1. Pass through the centroid of the cross section, and
2. Are parallel to, and in the same direction as, the global Cartesian X-, Y-, and Z-axes.

Note that DYCAST does not print the value of the seventh degree of freedom, i.e., the rate of twist θ, of the displacement node.

It is important to emphasize that although the nodal displacements, stresses, and strains are centroidal, the theory on which the subject beam element is based takes into account the eccentricity of the shear center with respect to the centroid in a rigorous manner.

1.3 CURVATURE & THE POINTER NODE

The "concavity/convexity" of the WARP curved beam element, whether it be two-noded or three-noded, is characterized in the DYCAST input data by the so-called pointer node and the element curvature, which is the inverse of the radius of curvature. The centroidal x-axis of the WARP element is assumed to be a circular arc, and, therefore, one radius of curvature for an entire element suffices. The numerical value of the curvature for a set of WARP elements is specified with the CURV card (see section 2.1), and it is always assumed to be positive or zero. Note that the curvature is that of the centroidal x-axis, i.e., the radius of curvature is measured from the center of curvature to the centroidal axis of the beam. (See Fig. 2.) It is recommended that the radius of curvature be greater than or equal to about 10 times the maximum dimension of the cross section.[1]

[1] This recommendation is based on considerations of curved beam theory rather than considerations of numerical stability. See, for example, Ref 2 and 5.

Whether the beam element is concave or convex depends on the location of the pointer node. Referring to Fig. 2a, denote the two end (displacement) nodes of a beam element as I and J. The WARP finite element is formulated such that:

1. End nodes I and J, circular arc $\overset{\frown}{IJ}$, the center of curvature of arc $\overset{\frown}{IJ}$, and the pointer node lie in the same plane, i.e., the local x-z plane defined by the local x- and z-element axes

2. Both the pointer node and the center of curvature of circular arc $\overset{\frown}{IJ}$ always lie on the same side of the chord $\overline{IJ}$, i.e., the straight line segment connecting nodes I and J. Stated differently, the pointer node must be placed on the center-of-curvature-side of chord $\overline{IJ}$.

The situation for a three-noded element is identical and is illustrated in Fig. 2b. Node K, the mid-node of the three-noded element, also lies in the x-z plane. Note from Fig. 2a and 2b that the local z-coordinate of the center of curvature is always negative.

The pointer node number for an element, whether two (displacement) noded or three (displacement) noded, is specified along with the connectivity data on the WARP cards, which are discussed in section 2.1. The location of the pointer node is arbitrary, as long as it satisfies the above two conditions 1 and 2. It is recommended that the pointer node be located at a reasonable distance from its element in order to avoid numerical problems. In actual computations with DYCAST, it has been observed that distances ranging from less than 3% of the overall length of the element to more than 150 times the length of the element have not caused any numerical problems. Note that the pointer node cannot be located on the chord connecting the two end nodes, i.e., the two end nodes and the pointer node cannot be colinear.

Finally, the WARP element in DYCAST is formulated such that the circular arc $\overset{\frown}{IJ}$, connecting end nodes I and J, cannot subtend an angle greater than 180 deg. (See Fig. 2a and 2b.)

For straight beams, one simply specifies a zero curvature on the CURV card discussed in section 2.1. The pointer node lies in the x-z half plane in which the local z-coordinate is negative. (See Fig. 3a and 3b.) Note again that the pointer node P cannot be located on the line segment $\overline{IJ}$ joining the two end nodes, i.e., the three nodes I, J, and P cannot be colinear.

2. WARP BEAM ELEMENT INPUT DATA

The input for the WARP beam element consists of a subset of DYCAST input supplemented by three additional new key word cards. Specific discussions on these input data are in the DYCAST user's manual, Ref 1, and in subsequent sections of this addendum. Listed below is a group of required and optional input types associated with the WARP beam element. Omitted in this table are key words that control print options and specification of nodal coordinates. Refer to the DYCAST user's manual, Ref 1, for details of those key words not discussed in detail below.

Group	Key Word	Required	Comments
A	DYNA	Yes	The first two input lines
A	STAT,EIGN	Yes	One of these two key words must be specified
A	LAMB*	Optional	Penalty parameter
C	WARP*	Yes	Element identification and connectivity
E	SPC	Yes	Single-point constraints
E	MPC	No	Multipoint constraints
E	APPL	No	Applied generalized displacements
H	CURV*	Yes	Element curvature
H	GSEC	Yes	Element section properties
H	MBM	Yes	Element material properties
I	CONC	No	Concentrated forces and moments at nodes
I	BMLO	No	Linearly varying line load on a beam element
I	PTME	No	Required for a linear static analysis

*New key word

2.1 NEW KEY WORDS FOR THE WARP ELEMENT

2.1.1 Key Word CURV

This is one of the three new key words/cards introduced into DYCAST by the new WARP beam finite element. It is part of the group H input data and specifies the curvature (which is the inverse of the radius of curvature) of a set of WARP beam elements. It is necessary only when WARP beam elements are used.

Card 1:

Format	Columns	Symbol	Comments
A4	1-4	CURV	Key word
	5		Blank
E15.0	6-20	CURV	Curvature of the beam element longitudinal centroidal x-axis
	21-80		Blank

Card(s) 2,3,4...:

The second and succeeding card(s) contain applicable member numbers.

Format	Columns	Symbol	Comments
16I5	1-80	MEM	The applicable member numbers for the above curvature are specified in fields of 5 (I5 format). The short form notation and any number of continuation cards may be used. A blank I5 field ends the specification of applicable members.

Notes

1. The curvature CURV is always greater than or equal to zero. For straight WARP beam elements, set CURV equal to zero, or leave the space blank.

2. Only one value of curvature for an entire beam element can be specified, because the longitudinal centroidal axis of the WARP element is assumed to be a circular arc that lies in one plane.

2.1.2 Key Word LAMB

This is one of three new key words/cards introduced into DYCAST by the new WARP beam finite element. It is part of the group A input data, and there is one LAMB card per finite element analysis. It specifies the normalized penalty parameter. The penalty parameter is used only in the computations of the WARP beam element, but even for WARP elements the LAMB card is optional. The default value in DYCAST for the normalized penalty parameter is 1.0, and the LAMB card in such a situation can be omitted. If another value is desired, the LAMB card would be required to input this alternative value.

Format	Columns	Symbol	Comments
A4	1-4	LAMB	Key word
	5		Blank
E15.0	6-20	PENLTY	Normalized penalty parameter
	21-80		Blank

Notes

1. The quantity PENLTY is the penalty parameter ϵ of Ref 2 normalized with respect to the axial stiffness EA of the beam element, where E is Young's modulus and A the cross-sectional area:

$$\text{PENLTY} = \frac{\epsilon}{EA}$$

Since PENLTY is the same for all WARP beam elements in a given analysis, the actual penalty parameter ϵ used to compute the stiffness matrix for a given beam element will vary from element to element, depending on the cross-sectional area and the Young's modulus of the element.

2. Results of finite element analyses with the WARP element tend to be insensitive to the magnitude of the variable PENLTY. In Ref 2, PENLTY was varied between 10^{-3} and 10^{+2}, and it was found that the natural frequencies are fairly insensitive to this variable. In section 3 of this report, the four example problems have been analyzed with PENLTY = 1.0. These problems were also reanalyzed with PENLTY = 0.01 and PENLTY = 100. All of the results, i.e., the nodal displacements, stresses, strains, natural frequencies, and eigenvector components, were found to differ by less than one percent from the results of the original analyses with PENLTY = 1.0. In fact, the discrepancies in the lowest six natural frequencies of example problem 3 in this report (refer to Table 1) were less than 0.005%.

2.1.3 Key Word WARP

This is one of three new key words/cards introduced into DYCAST by the new WARP beam finite element. It is part of the group C input data, and it specifies element connectivity. One such card is required for each element. As in the case of all of DYCAST's other element types, the WARP elements in a given finite element model need not be consecutively numbered, i.e., one can "skip" element identification numbers in the model. Also, the WARP cards do not have to be ordered in the input data according to monotonically increasing element identification numbers.

Format	Columns	Symbol	Comments
A4	1-4	WARP	Key word
	5-10		Blank
I5	11-15	MEM	Element identification number
I5	16-20	NODEI	Node identification number of the end (displacement) node that is located at $x = -\frac{l}{2}$ according to the local beam element coordinate system, l being the arc length of the element.
I5	21-25	NODEJ	Node identification number of the end (displacement) node that is located at $x = +\frac{l}{2}$
I5	26-30	NODEK	For an element with three displacement nodes, this is the node identification number of the mid-node, located at x = 0. For an element with two displacement nodes, set this variable equal to zero, or leave this space blank.
I5	31-35	NODEL	This is the node identification number of the pointer node of the element. It is used to orient the local x-z plane of the element in the global Cartesian XYZ coordinate system. See the section "Curvature & the Pointer Node."
	36-80		Blank

Notes

1. The number of displacement nodes per WARP element can be either two or three. In defining the element connectivity of a two-noded element, the node numbers NODEI and NODEJ of the two end nodes and the node number of the pointer node NODEL are specified. In the case of a three-noded element, the identification numbers of the two end nodes, the mid-node, and the pointer node are specified. Which of these two options is used is solely determined by the variable NODEK. Thus, any one finite element model of a structural system can consist of both two-noded and three-noded DYCAST WARP elements.

2. Several WARP beam elements may all have the same node as their pointer node, provided that their local x- and z-axes lie in the same plane and their centers of curvature are thereby properly specified. Note that the pointer node of a WARP element may be one of the end nodes or the mid-node of another element.

3. In the case of three displacement nodes per element, the DYCAST program does not use, in its internal computations, the global Cartesian X-,Y-, and Z- coordinates of the mid-node K, which are inputted with the GRDX, GRDY, GRDZ, or GRID cards. This is possible because the value of the curvature is required for every element. Thus, if no other finite elements connect to the mid-node K of a particular WARP element, and if this mid-node K is not used as the pointer node of another element, then the user need not be concerned about the exactness of the X-, Y-, and Z-coordinates of the mid-node. Note, however, that the node identification number of the mid-node must always be inputted correctly on the WARP card.

2.2 COMMENTS ON USAGE OF EXISTING KEY WORDS

2.2.1 Key Words DYNA, STAT, & EIGN

The input format of these three key words has not been modified for the WARP beam element. However, note that the WARP element can be used only for either:

1. Linear, elastic, static analysis with one applied external loading, or
2. Linear, elastic, free vibration analysis.

Thus, when using WARP elements, the DYNA cards in the DYCAST input must always be accompanied by either the STAT card (for static analysis) or the EIGN card (for free vibration analysis).

In static analyses with the WARP element, DYCAST outputs the nodal deflections and angular rotations with respect to the global coordinate system, and the element nodal stress resultants, strains, curvature changes, twist, etc. with respect to the local element coordinate systems.

In free-vibration analyses with the WARP element, DYCAST outputs the natural frequencies of vibration of the finite element model and the corresponding eigenvectors, the components of which are the nodal displacement degrees of freedom with respect to the global coordinate system.

2.2.2 Key Word BMLO

The input format of this key word has not been modified for the WARP beam element.

Notes

1. For the three-noded WARP element, only the force factors for the two end (displacement) nodes are inputted. The force factor for the mid-node is internally interpolated by DYCAST by considering that the distributed line load varies linearly, not quadratically, along the length of the beam.

2. In the case of WARP beam elements, the BMLO cards can be used only in linear static analyses. In this situation, the two PTME cards, which specify the (constant) magnitude of the loading, must be used with a nonzero value for the force at the initial solution step (time = 0).

3. It should be emphasized that the line load one specifies with the BMLO card passes through the centroidal axis of the beam and not the line of shear centers. Note also that the line load specified is always perpendicular to the (curved) centroidal axis.

2.2.3 Key Word CONC

The input format of this key word has not been modified for the WARP beam element.

Notes

1. In the case of the WARP beam element, the CONC cards can be used only in linear static analyses. In this situation, the two PTME cards, which specify the (constant) magnitude of the loading, must be used with a nonzero value for the force at the initial solution step (time = 0).

2. The input nodal forces and moments are oriented with respect to the global Cartesian X-, Y-, and Z-axes.

2.2.4 Key Word GSEC

The key words for the 13 pre-formed special cross sections of DYCAST, namely the SREC, SCIR, HREC, HCIR, LSEC, TSEC, ISEC, ZSEC, ZSCR, CSEC, HSEC, TWD, and LSEG cross sections, can not be used with the WARP beam element. Only the GSEC card, which is for general thin-walled cross sections, can be used with this element. The input format on the GSEC cards for the WARP element is identical to the case of the DYCAST BEAM element. However, there are some restrictions associated with using the GSEC card for WARP elements, as described in the following notes.

Notes

1. DYCAST locates the centroid, or center of gravity, of the cross section and computes (internally) all of the cross-sectional properties with respect to the local, centroidal, orthogonal y- and z-axes. These properties, which are used in the computation of the element stiffness and mass matrices, include the area, the moments and product of inertia, the St. Venant torsion constant, the warping constant (I_ω), the first sectorial moment (S_ω), and the sectorial linear statical moments ($I_{\omega y}$ and $I_{\omega z}$). Thus, DYCAST automatically moves the origin of the arbitrary y- and z-axes that are specified by the user on the GSEC cards to the centroid of the cross section before the actual finite element analysis is begun. Consequently, the user does not have any choice in the location of the longitudinal x-axis, and the nodal degrees of freedom in the DYCAST output data will always refer to the centroidal nodal deflections, rotations, and stress resultants.

   However, DYCAST does not change the orientation of the (local) y- and z-axes with respect to the cross section. Their orientation remains that initially specified by the user on the GSEC card. Thus, DYCAST will perform the finite element computations for a particular WARP element in the principal xyz coordinate system if and only if the user-specified orientation of the y- and z-axes is principal. Whether the element local coordinate system is principal or not, the global nodal displacement degrees of freedom that DYCAST prints out remain in the global Cartesian XYZ coordinate system, i.e., they are "invariant" with respect to the local element coordinate systems.

   Note that even though centroidal cross-sectional properties, which may or may not be principal, are used in the finite element computations, DYCAST prints out anyway, for the user's information, the (local) y- and z-coordinates of the shear center, the warping constant with respect to the shear center, and the principal (centroidal) moments of inertia.

Since DYCAST performs all of the cross-section-related computations for the WARP element internally, the first two GSEC cards must remain blank from columns 5 to 80. Any numerical value in these columns is ignored by the WARP element.

2. The third GSEC card must indicate that the cross section is open because the WARP beam element has been formulated for open cross sections (Ref 2). In addition, DYCAST does not have the capability of computing the quantities I_{ω}, $I_{\omega y}$, $I_{\omega z}$, and S_{ω} for a closed cross section.

3. A GSEC end point, or node, is recommended at sharp corners in the contour of the cross section (such as the intersection of the web and the flange of a channel beam) and at junctions (such as the intersection of the web and the flange of an I-beam). However, including more than one GSEC segment in a <u>straight</u> section of the contour will not increase the accuracy of the computations for the cross-sectional beam properties.

4. Two straight-line segments that are adjacent to each other in the cross section need not be <u>numbered</u> consecutively, nor do two adjacent segments have to be <u>listed</u> consecutively in the GSEC input data. Any segment can be listed as the first in the GSEC data. However, note that the first end point of the first segment listed in the GSEC input data, is the point that is taken by DYCAST as the location, on the contour of the cross section, where the contour warping function is zero. Thus, the user can freely choose this zero location simply through the ordering/numbering scheme on the GSEC cards.

5. The number of Gaussian integration points per straight-line segment, NINT, on the fifth GSEC card is irrelevant because the material of the WARP beam element is restricted to being linear and elastic. Any dummy value between NINT = 2.0 and NINT = 8.0 is acceptable.

6. Note that the cross section in any one WARP beam element does not vary along the length of the beam.

7. Avoid cross sections in which any of the cross-sectional properties is identically zero. One example of such a situation is the thin rectangular section, i.e., one web and no flanges. If the local y- and z-axes are specified to be principal, one of the moments of inertia and the warping constant will be computed by DYCAST as identically zero. One should therefore also be cautious about using zero thicknesses for the segments of the cross section.

8. In summary, the only numerical quantities needed for a particular open cross section of a given set of WARP beam elements are the number of straight-line segments per cross section, the y- and z-coordinates of the two end points of every segment, and the thickness of each segment.

2.2.5 Key Word MBM

The input format of this key word has not been modified for the WARP beam element.

Notes

1. The material in the WARP beam element formulation is assumed to be linear, elastic, and isotropic. Thus, only Young's modulus E and the Poisson ratio υ are required. The Poisson ratio is used by the WARP element to calculate internally a shear modulus G, i.e.,

$$G = \frac{E}{2(1+\upsilon)}$$

The shear modulus is important because, unlike finite elements based on elementary beam theory, transverse shear deformations are included in the WARP element formulation.

2. The material in any one WARP beam element is assumed to be homogeneous, but obviously the material properties can vary from WARP element to WARP element.

2.2.6 Key Words SPC, MPC, & APPL

When WARP beam elements are specified in the input data, every node in the finite element model is automatically assigned seven degrees of freedom (DOF), rather than the usual six or three DOF's. This is true whether or not a node has WARP elements connecting to it. The seven degrees of freedom are:

- DOF 1 = Translational displacement in the global X-direction
- DOF 2 = Translational displacement in the global Y-direction
- DOF 3 = Translational displacement in the global Z-direction
- DOF 4 = Angle of rotation about the global X-axis
- DOF 5 = Angle of rotation about the global Y-axis
- DOF 6 = Angle of rotation about the global Z-axis
- DOF 7 = Rate of twist θ in the local coordinate system of the WARP element.

Therefore, what distinguishes the WARP element is that it contributes stiffness to the seventh DOF, whereas all of the other finite elements in the DYCAST library do not.

Due to the new seventh DOF, the input formats for key words SPC, MPC, and APPL have been slightly modified, as described in the following notes.

Notes

1. A seven-digit sequence in columns 6 to 12, instead of just six or three digits, is now required on the SPC card in the order of the above list, i.e., DOF 1 in column 6, DOF 2 in column 7, ..., DOF 7 in column 12. On the MPC card, the dependent and independent degree-of-freedom numbers now range from 1 to 7 instead of 1 to 6. If a node has neither an SPC nor MPC constraint assigned to it, DYCAST by default frees all seven degrees-of-freedom at that node. This situation would correspond to the following seven-digit sequence on the SPC card: 1111111.

2. On the APPL card, the degree-of-freedom number at which displacement is applied, located in columns 11 to 15, now ranges from 1 to 7 instead of 1 to 6.

3. Finally, note that the key word ACEL is not applicable to the present version of the WARP element.

3. EXAMPLE PROBLEMS

For the purpose of checking the new WARP beam element and the software that acts as the interface between the original version of DYCAST and the new finite element, the example problems from Ref 2 have been used. Good agreement has been obtained with all of the free-vibration analyses of Ref 2.

To illustrate the use of the WARP element, the following two example probems from Ref 2 have been chosen:

1. The cantilever beam with a symmetric channel cross section;
2. The semicircular, clamped arch with a symmetric Z-cross section.

The problems are illustrated in Fig. 4 and 5. Note that both the beam and the arch are arbitrarily oriented in the global Cartesian XYZ coordinate system. The dimensions of the cross sections are given in Fig. 2 of Ref 2 for the channel section and in Fig. 8 of this reference for the Z-section.

For the channel section, each of the two flanges is 2.54 cm long and 0.0635 cm thick, and the web is 1.27 cm long and 0.0635 cm thick. The material properties for the cantilever beam are as follows: Young's modulus $E = 6.895 \times 10^{10}$ Pa, Poisson's ratio $\upsilon = 0.32$, and the mass density $\rho = 2600$ kg/m^3. The normalized penalty parameter, identified as the FORTRAN variable PENLTY (refer to section 2.1), is equal to one.

For the Z-section, each of the two flanges is 2.54 cm long and 0.60 cm thick, and the web is 5.08 cm long and 0.60 cm thick. The material properties of the semicircular arch are as follows: Young's modulus $E = 7.17 \times 10^{10}$ Pa, Poisson's ratio $\upsilon = 0.30$, and the mass density $\rho = 2768$ kg/m^3. The normalized penalty parameter, identified as the FORTRAN variable PENLTY (refer to section 2.1), is equal to one.

A uniform line load of magnitude 1.0 was applied to the cantilever beam, and a linear, static, finite element analysis was performed with the updated version of DYCAST. The line load passed through the line of centroids of the channel cross sections and was directed parallel to the web. This eccentricity of the applied loading with respect to the line of shear centers induced both flexural and torsional behavior of the cantilever. In the case of the semicircular arch, a free-vibration analysis was

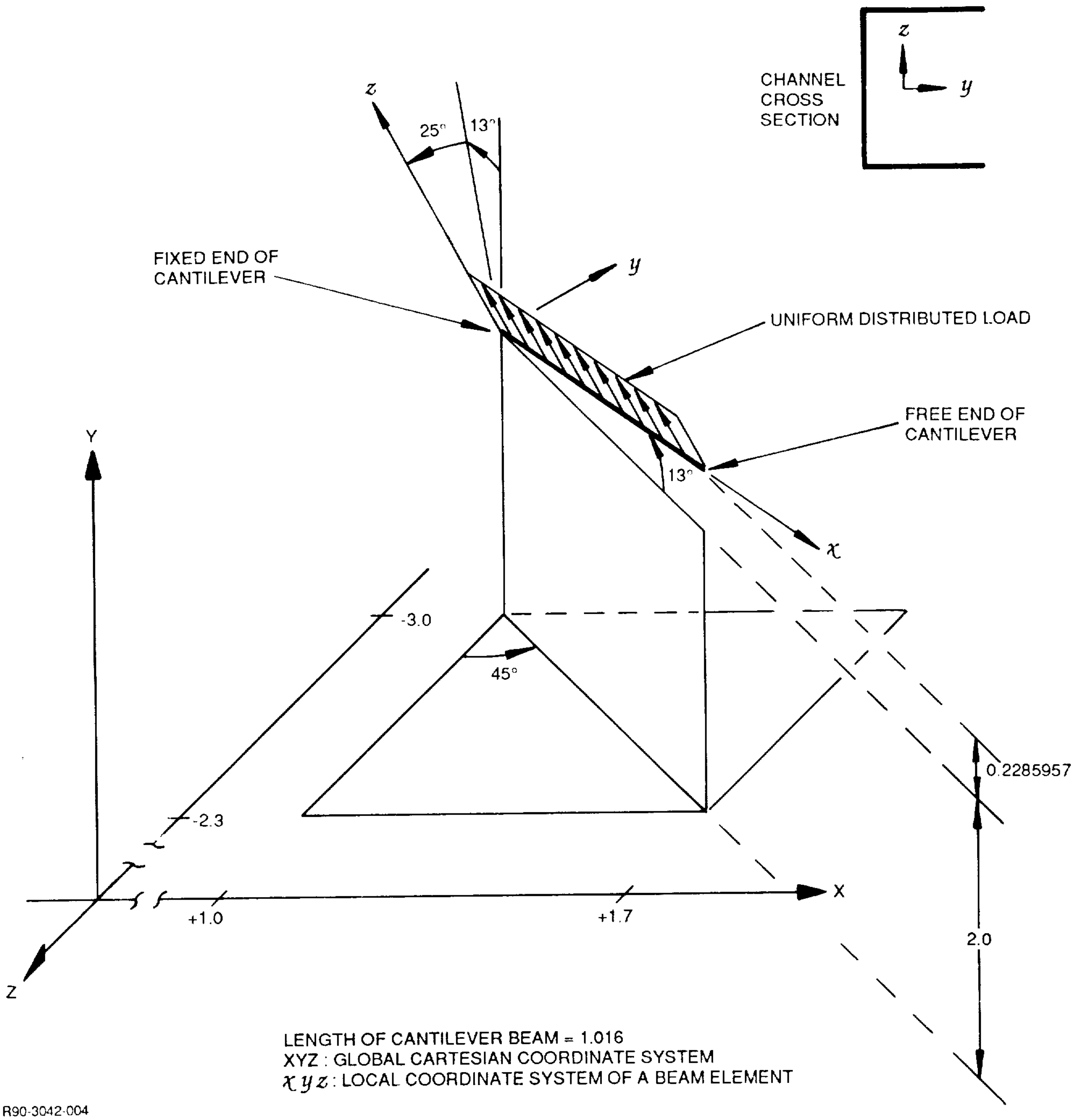


Fig. 4 Cantilever Beam Problem

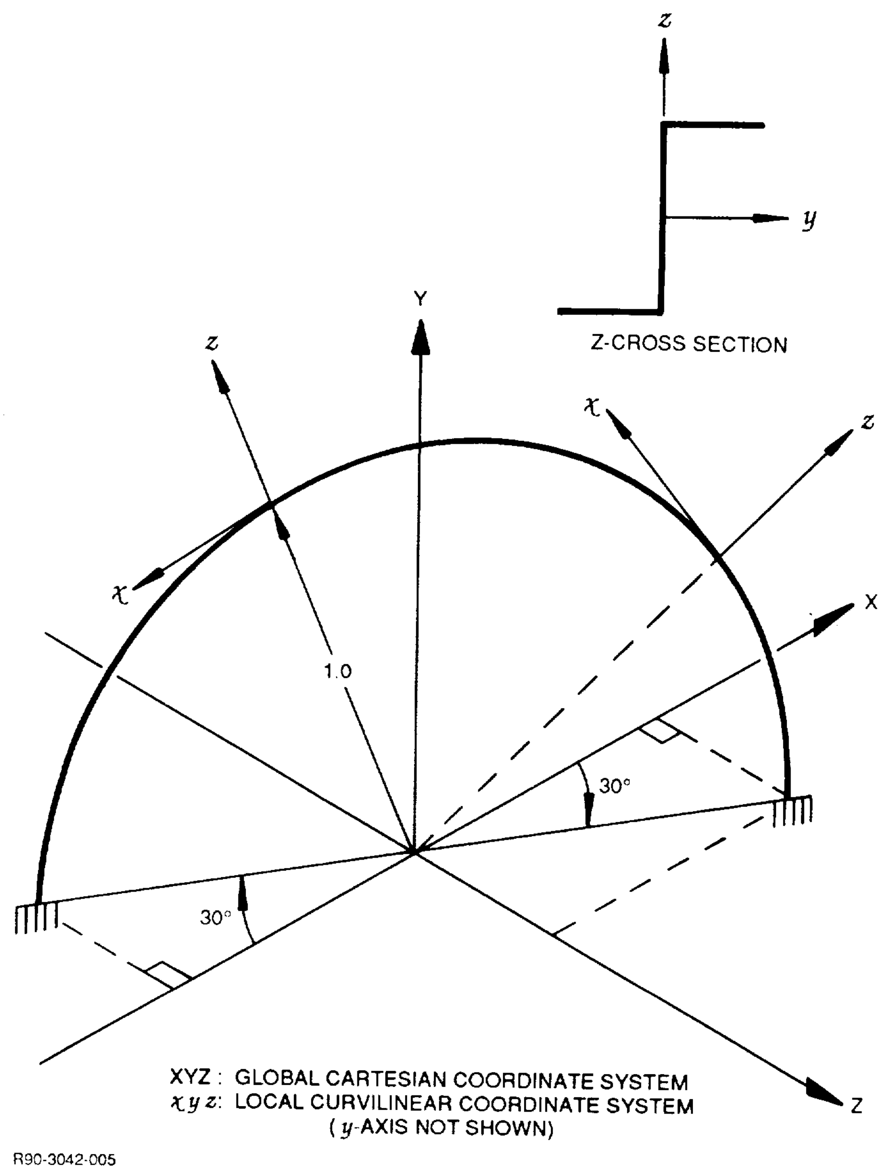


Fig. 5 Semicircular Clamped Arch Problem

performed, resulting in the natural frequencies of vibration and the corresponding eigenvectors.

In both problems, two finite element models were used:

1. Eight WARP beam elements, each element with three displacement nodes (and two stress nodes), all elements of equal length;

2. Sixteen WARP beam elements, each element with two displacement nodes (and one stress node), all elements of equal length.

In the case of the static analysis of the cantilever, excellent agreement was obtained between the displacements, stresses, and strains of the analytical beam theory solution of Ref. 5 and those of the two DYCAST finite element solutions (i.e., both the 8-element and 16-element analyses). The deflection vectors and the rotation vectors of the midpoint and the free end of the cantilever, calculated with DYCAST, were transformed into the local xyz coordinate system of the elements. The resulting z-deflections, which were in the direction of the uniform loading, and the resulting torsional angles of rotation were compared to the analytically obtained values. The maximum discrepancy in the case of the finite element model with 8 three-noded elements was 0.03%, and in the case of 16 two-noded elements it was 0.20%.

Good results were also obtained for the free-vibration analysis of the semicircular arch. The lowest six natural frequencies of the semicircular arch were calculated by DYCAST with the eight three-noded WARP beam element model. On comparison of these frequencies with those of Noor et al. (Ref 2), which were obtained with a finite element model of the arch consisting of shell elements, it was found that the discrepancy ranged from 1.0% to 4.8%. (See Table 1.) On comparison of these same DYCAST frequencies to those in Ref. 2 obtained with a beam element model, the discrepancy of the frequencies ranged from 0.18% to 2.6%. These latter discrepancies are most likely due to differences in discretization of the finite element models of the arch. Unfortunately, the number of thin-walled beam elements used and the number of nodes per element have not been disclosed in Ref 2.

Table 1 also displays the frequencies calculated by DYCAST with the 16 WARP element model, in which there are two displacement nodes per element. The frequencies are consistently higher in numerical value than those obtained in the case

TABLE 1
NATURAL FREQUENCIES OF VIBRATION OF CLAMPED, SEMICIRCULAR ARCH WITH SYMMETRIC Z-SECTION

Mode Number	Natural Frequency (Hz) Obtained with Shell Finite Element Model of Ref. 1	Natural Frequency (Hz) Obtained with Thin-Walled Beam Element Model of Ref. 1	Natural Frequency (Hz) Obtained with Finite Element Program DYCAST*	
			Eight Three-Noded Thin-Walled Beam Elements	Sixteen Two-Noded Thin-Walled Beam Elements
1	10.70	11.10	11.12	11.47
2	24.02	24.17	24.25	25.40
3	51.72	52.13	52.65	56.59
4	63.02	64.41	64.72	68.10
5	91.41	92.12	94.16	104.1
6	128.7	131.5	134.9	148.0

* Refer to Fig. 5

of three displacement nodes per element, indicating a stiffer finite element model of the arch. This is to be expected, for piecewise quadratic displacement fields should be more accurate than piecewise linear fields, provided the number of displacement nodes is the same in both situations.

Given as Tables 2 through 5 are the DYCAST input decks for the four example problems described above:

1. The static analysis of the cantilever problem with 8 three-noded WARP elements (Table 2)
2. The static analysis of the cantilever problem with 16 two-noded WARP elements (Table 3)
3. The semicircular arch eigenproblem with 8 three-noded WARP elements (Table 4)
4. The semicircular arch eigenproblem with 16 two-noded WARP elements (Table 5).

Table 2 Static Cantilever Problem: Three-Noded Elements (Sheet 1 of 3)

```
      DYCAST EXAMPLE PROBLEM ONE.
$     STATIC LINEAR ANALYSIS OF CANTILEVER BEAM;
          SYMMETRIC CHANNEL CROSS SECTION;
$         UNIFORM LINE LOAD ACTING THROUGH THE CENTROID;
$         EIGHT THREE-NODED WARP ELEMENTS.
$
$
DYNA  1000    1    1
DYNA      0.0             0.0              1.0
$EIGN          10.0          2.0
STAT
$SCAN
SEND
NODE     1  -19
SEND
WARP           1    1    3    2    18
WARP           2    3    5    4    19
WARP           3    5    7    6    18
WARP           4    7    9    8    18
WARP           5    9   11   10    19
WARP           6   11   13   12    19
WARP           7   13   15   14    18
WARP           8   15   17   16    19
SEND
GRDX   1.0               1
GRDX   1.04375           2
GRDX   1.0875            3
GRDX   1.13125           4
GRDX   1.175             5
GRDX   1.21875           6
GRDX   1.2625            7
GRDX   1.30625           8
GRDX   1.35              9
GRDX   1.39375          10
GRDX   1.4375           11
GRDX   1.48125          12
GRDX   1.525            13
GRDX   1.56875          14
GRDX   1.6125           15
GRDX   1.65625          16
GRDX   1.7              17
GRDX   5.430261         18
GRDX   1.7443026        19
GRDY   2.0               1
GRDY   2.0142872         2
GRDY   2.0285745         3
GRDY   2.0428617         4
GRDY   2.0571489         5
GRDY   2.0714361         6
GRDY   2.0857234         7
GRDY   2.1000106         8
```

Table 2 Static Cantilever Problem: Three-Noded Elements (Sheet 2 of 3)

```
GRDY    2.1142978         9
GRDY    2.128585         10
GRDY    2.1428723        11
GRDY    2.1571595        12
GRDY    2.1714467        13
GRDY    2.185734         14
GRDY    2.2000212        15
GRDY    2.2143084        16
GRDY    2.2285957        17
GRDY   -6.830698         18
GRDY    2.1402888        19
GRDZ   -3.0               1
GRDZ   -2.95625           2
GRDZ   -2.9125            3
GRDZ   -2.86875           4
GRDZ   -2.825             5
GRDZ   -2.78125           6
GRDZ   -2.7375            7
GRDZ   -2.69375           8
GRDZ   -2.65              9
GRDZ   -2.60625          10
GRDZ   -2.5625           11
GRDZ   -2.51875          12
GRDZ   -2.475            13
GRDZ   -2.43125          14
GRDZ   -2.3875           15
GRDZ   -2.34375          16
GRDZ   -2.3              17
GRDZ   -4.546464         18
GRDZ   -2.3154646        19
SEND
SPC  0000000              1    18    19
SPC  1111111              2     4     6     8    10    12    14    16
SPC  1111111              3     5     7     9    11    13    15    17
SEND
MBM      6.895   E+10     .32            0.0             0.0           3.0    E+1
           2600.
   1    -8
CURV       0.0
   1    -8
GSEC
GSEC
GSEC OPEN      3
GSEC     1     .0254          .00635         0.0            .00635
GSEC           6.35  E-04     3.0
GSEC     2     0.0            0.00635        0.0            -0.00635
GSEC           6.35  E-04     3.0
GSEC     3     0.0254         -0.00635       0.0            -0.00635
GSEC           6.35  E-04     3.0
   1    -8
```

Table 2 Static Cantilever Problem: Three-Noded Elements (Sheet 3 of 3)

```
SEND
$CONC     17     0.0          0.0          1.0
$CONC
BMLO             0.0          0.0          1.0          1.0
    1    -8
PTME             1.0          0.0
PTME             1.0          1.0
SEND
STOP
```

Table 3 Static Cantilever Problem: Two-Noded Elements (Sheet 1 of 3)

```
       DYCAST EXAMPLE PROBLEM TWO.
$      STATIC LINEAR ANALYSIS OF CANTILEVER BEAM;
$           SYMMETRIC CHANNEL CROSS SECTION;
$           UNIFORM LINE LOAD ACTING THROUGH THE CENTROID;
$           SIXTEEN TWO-NODED WARP ELEMENTS.
$
DYNA   1000    1    1
DYNA       0.0              0.0              1.0
$EIGN            10.0          2.0
STAT
$SCAN
SEND
NODE      1   -19
SEND
WARP           1    1    2         18
WARP           2    2    3         19
WARP           3    3    4         19
WARP           4    4    5         18
WARP           5    5    6         19
WARP           6    6    7         18
WARP           7    7    8         18
WARP           8    8    9         18
WARP           9    9   10         19
WARP          10   10   11         18
WARP          11   11   12         19
WARP          12   12   13         19
WARP          13   13   14         19
WARP          14   14   15         18
WARP          15   15   16         18
WARP          16   16   17         19
SEND
GRDX    1.0                 1
GRDX    1.04375             2
GRDX    1.0875              3
GRDX    1.13125             4
GRDX    1.175               5
GRDX    1.21875             6
GRDX    1.2625              7
GRDX    1.30625             8
GRDX    1.35                9
GRDX    1.39375            10
GRDX    1.4375             11
GRDX    1.48125            12
GRDX    1.525              13
GRDX    1.56875            14
GRDX    1.6125             15
GRDX    1.65625            16
GRDX    1.7                17
GRDX    5.430261           18
GRDX    1.7443026          19
```

Table 3 Static Cantilever Problem: Two-Noded Elements (Sheet 2 of 3)

```
GRDY    2.0              1
GRDY    2.0142872        2
GRDY    2.0285745        3
GRDY    2.0428617        4
GRDY    2.0571489        5
GRDY    2.0714361        6
GRDY    2.0857234        7
GRDY    2.1000106        8
GRDY    2.1142978        9
GRDY    2.128585        10
GRDY    2.1428723       11
GRDY    2.1571595       12
GRDY    2.1714467       13
GRDY    2.185734        14
GRDY    2.2000212       15
GRDY    2.2143084       16
GRDY    2.2285957       17
GRDY   -6.830698        18
GRDY    2.1402888       19
GRDZ   -3.0              1
GRDZ   -2.95625          2
GRDZ   -2.9125           3
GRDZ   -2.86875          4
GRDZ   -2.825            5
GRDZ   -2.78125          6
GRDZ   -2.7375           7
GRDZ   -2.69375          8
GRDZ   -2.65             9
GRDZ   -2.60625         10
GRDZ   -2.5625          11
GRDZ   -2.51875         12
GRDZ   -2.475           13
GRDZ   -2.43125         14
GRDZ   -2.3875          15
GRDZ   -2.34375         16
GRDZ   -2.3             17
GRDZ   -4.546464        18
GRDZ   -2.3154646       19
SEND
SPC   0000000           1    18    19
SPC   1111111           2     4     6     8    10    12    14    16
SPC   1111111           3     5     7     9    11    13    15    17
SEND
MBM       6.895    E+10     .32           0.0           0.0          3.0    E+1
            2600.
   1   -16
CURV        0.0
   1   -16
GSEC
GSEC
```

Table 3 Static Cantilever Problem: Two-Noded Elements (Sheet 3 of 3)

```
GSEC  OPEN      3
GSEC       1     .0254              .00635          0.0          .00635
GSEC             6.35   E-04        3.0
GSEC       2     0.0                0.00635         0.0          -0.00635
GSEC             6.35   E-04        3.0
GSEC       3     0.0254             -0.00635        0.0          -0.00635
GSEC             6.35   E-04        3.0
    1   -16
SEND
$CONC     17      0.0                0.0            1.0
$CONC
BMLO             0.0                0.0             1.0          1.0
    1   -16
PTME             1.0                0.0
PTME             1.0                 1.0
SEND
STOP
```

Table 4 Semicircular Arch Eigenproblem: Three-Noded Elements (Sheet 1 of 3)

```
      DYCAST EXAMPLE PROBLEM THREE.
$     FREE VIBRATION ANALYSIS OF SEMICIRCULAR CLAMPED ARCH;
$          SYMMETRIC Z-CROSS SECTION;
$          EIGHT THREE-NODED WARP ELEMENTS.
$
DYNA 00000    1    1
DYNA      0.0               0.0              1.0
EIGN          10.0          2.0
$STAT
$SCAN
LAMB   1.00
SEND
NODE     1  -20
SEND
WARP          1    1    3    2   18
WARP          2    3    5    4   19
WARP          3    5    7    6   18
WARP          4    7    9    8   20
WARP          5    9   11   10   19
WARP          6   11   13   12   20
WARP          7   13   15   14   18
WARP          8   15   17   16   19
SEND
GRDX   0.8660254        1
GRDX   0.8493849        2
GRDX   0.8001031        3
GRDX   0.7200738        4
GRDX   0.6123723        5
GRDX   0.4811379        6
GRDX   0.3314135        7
GRDX   0.1689531        8
GRDX   0.0              9
GRDX  -0.1689531       10
GRDX  -0.3314135       11
GRDX  -0.4811379       12
GRDX  -0.6123723       13
GRDX  -0.7200738       14
GRDX  -0.8001031       15
GRDX  -0.8493849       16
GRDX  -0.8660254       17
GRDX   0.2165063       18
GRDX  -0.7794228       19
GRDX   0.0             20
GRDY   0.0              1
GRDY   0.1950903        2
GRDY   0.3826834        3
GRDY   0.5555702        4
GRDY   0.7071067        5
GRDY   0.8314696        6
GRDY   0.9238795        7
```

Table 4 Semicircular Arch Eigenproblem: Three-Noded Elements (Sheet 2 of 3)

```
GRDY    0.9807852          8
GRDY    1.0                9
GRDY    0.9807852         10
GRDY    0.9238795         11
GRDY    0.8314696         12
GRDY    0.7071067         13
GRDY    0.5555702         14
GRDY    0.3826834         15
GRDY    0.1950903         16
GRDY    0.0               17
GRDY    0.5               18
GRDY   -4.4               19
GRDY    0.99              20
GRDZ    0.5                1
GRDZ    0.4903926          2
GRDZ    0.4619397          3
GRDZ    0.4157348          4
GRDZ    0.3535533          5
GRDZ    0.2777851          6
GRDZ    0.1913417          7
GRDZ    0.0975451          8
GRDZ    0.0                9
GRDZ   -0.0975451         10
GRDZ   -0.1913417         11
GRDZ   -0.2777851         12
GRDZ   -0.3535533         13
GRDZ   -0.4157348         14
GRDZ   -0.4619397         15
GRDZ   -0.4903926         16
GRDZ   -0.5               17
GRDZ    0.125             18
GRDZ   -0.45              19
GRDZ    0.0               20
SEND
SPC   0000000              1    17    18    19    20
SPC   1111111              2     4     6     8    10    12    14    16
SPC   1111111              3     5     7     9    11    13    15    17
SPC   0000000             17
SEND
MBM      7.170    E+10      .30            0.0            0.0          3.0    E+1
           2768.
    1    -8
CURV       1.0
    1    -8
GSEC
GSEC
GSEC  OPEN       4
GSEC      1      -0.0254          -0.0254          0.0            -0.0254
GSEC              0.006            3.0
GSEC      2       0.0             -0.0254          0.0             0.0
```

Table 4 Semicircular Arch Eigenproblem: Three-Noded Elements (Sheet 3 of 3)

```
GSEC          0.006        3.0
GSEC     3     0.0         0.0            0.0            0.0254
GSEC          0.006        3.0
GSEC     4    0.0          0.0254         0.0254         0.0254
GSEC          0.006        3.0
    1   -8
SEND
CONC    17    0.0         0.0            1.0
CONC
PTME          1.0         0.0
PTME          1.0          1.0
SEND
STOP
```

Table 5 Semicircular Arch Eigenproblem: Two-Noded Elements (Sheet 1 of 3)

```
      DYCAST EXAMPLE PROBLEM FOUR.
$     FREE VIBRATION ANALYSIS OF SEMICIRCULAR CLAMPED ARCH;
$          SYMMETRIC Z-CROSS SECTION;
$          SIXTEEN TWO-NODED WARP ELEMENTS.
$
DYNA 00000     1     1
DYNA      0.0                0.0               1.0
EIGN           8.0                2.0
$STAT
$SCAN
SEND
NODE     1  -20
SEND
WARP           1     1     2          18
WARP           2     2     3          20
WARP           3     3     4          19
WARP           4     4     5          18
WARP           5     5     6          18
WARP           6     6     7          18
WARP           7     7     8          20
WARP           8     8     9          20
WARP           9     9    10          20
WARP          10    10    11          19
WARP          11    11    12          18
WARP          12    12    13          20
WARP          13    13    14          18
WARP          14    14    15          19
WARP          15    15    16          19
WARP          16    16    17          19
SEND
GRDX   0.8660254           1
GRDX   0.8493849           2
GRDX   0.8001031           3
GRDX   0.7200738           4
GRDX   0.6123723           5
GRDX   0.4811379           6
GRDX   0.3314135           7
GRDX   0.1689531           8
GRDX   0.0                 9
GRDX  -0.1689531          10
GRDX  -0.3314135          11
GRDX  -0.4811379          12
GRDX  -0.6123723          13
GRDX  -0.7200738          14
GRDX  -0.8001031          15
GRDX  -0.8493849          16
GRDX  -0.8660254          17
GRDX   0.2165063          18
GRDX  -0.7794228          19
GRDX   0.0                20
```

Table 5 Semicircular Arch Eigenproblem: Two-Noded Elements (Sheet 2 of 3)

```
GRDY    0.0             1
GRDY    0.1950903       2
GRDY    0.3826834       3
GRDY    0.5555702       4
GRDY    0.7071067       5
GRDY    0.8314696       6
GRDY    0.9238795       7
GRDY    0.9807852       8
GRDY    1.0             9
GRDY    0.9807852      10
GRDY    0.9238795      11
GRDY    0.8314696      12
GRDY    0.7071067      13
GRDY    0.5555702      14
GRDY    0.3826834      15
GRDY    0.1950903      16
GRDY    0.0            17
GRDY    0.5            18
GRDY   -4.4            19
GRDY    0.99           20
GRDZ    0.5             1
GRDZ    0.4903926       2
GRDZ    0.4619397       3
GRDZ    0.4157348       4
GRDZ    0.3535533       5
GRDZ    0.2777851       6
GRDZ    0.1913417       7
GRDZ    0.0975451       8
GRDZ    0.0             9
GRDZ   -0.0975451      10
GRDZ   -0.1913417      11
GRDZ   -0.2777851      12
GRDZ   -0.3535533      13
GRDZ   -0.4157348      14
GRDZ   -0.4619397      15
GRDZ   -0.4903926      16
GRDZ   -0.5            17
GRDZ    0.125          18
GRDZ   -0.45           19
GRDZ    0.0            20
SEND
SPC   0000000           1   17   18   19   20
SPC   1111111           2    4    6    8   10   12   14   16
SPC   1111111           3    5    7    9   11   13   15   17
SPC   0000000          17
SEND
MBM     7.170   E+10    .30          0.0          0.0          3.0   E+1
          2768.
    1   -16
CURV      1.0
```

Table 5 Semicircular Arch Eigenproblem: Two-Noded Elements (Sheet 3 of 3)

```
    1   -16
GSEC
GSEC
GSEC OPEN       4
GSEC      1      -0.0254         -0.0254         0.0            -0.0254
GSEC             0.006            3.0
GSEC      2       0.0            -0.0254         0.0             0.0
GSEC             0.006            3.0
GSEC      3       0.0             0.0            0.0             0.0254
GSEC             0.006            3.0
GSEC      4      0.0              0.0254         0.0254          0.0254
GSEC             0.006            3.0
    1   -16
SEND
CONC     17      0.0             0.0             1.0
CONC
PTME             1.0             0.0
PTME             1.0              1.0
SEND
STOP
```

4. REFERENCES

1. Pifko, A. B.; Winter, R.; and Ogilive, P. L. : DYCAST--A Finite Element Program for the Crash Analysis of Structures. NASA CR-4040, 1987.

2. Noor, A. K.; Peters, J. M.; and Min, B. J. : Mixed Finite Element Models for Free Vibrations of Thin-Walled Beams. NASA TP-2868, Feb. 1989.

3. Noor, A. K.; and Peters, J. M. : Mixed Models and Reduced/Selective Integration Displacement Models for Nonlinear Analysis of Curved Beams. International Journal for Numerical Methods in Engineering. Vol. 17, 1981, pp 615-631.

4. Gjelsvik, A. : The Theory of thin Walled Bars. John Wiley & Sons, 1981.

5. Roark, R. J.; and Young, W. C. : Formulas for Stress and Strain. 5th ed., McGraw-Hill, 1975.

Report Documentation Page

NASA — National Aeronautics and Space Administration

1. Report No. NASA CR-4362	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle Addendum to the DYCAST User's Manual Describing the Curved, Warp Beam Finite Element		5. Report Date May 1991
		6. Performing Organization Code
7. Author(s) L. M. Kutt and A. B. Pifko		8. Performing Organization Report No. RE-780
		10. Work Unit No. 510-02-12-03
9. Performing Organization Name and Address Lockheed Missiles & Space Company, Inc. 3251 Hanover Street Palo Alto, CA 94304		11. Contract or Grant No. NAS1-18444, Task 8
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Langley Research Center Hampton, VA 23665-5225		13. Type of Report and Period Covered Contractor Report
		14. Sponsoring Agency Code

15. Supplementary Notes

L. M. Kutt and A. B. Pifko: Grumman Corporate Research Center, Bethpage, New York 11714-3580.

This report was prepared by Grumman Corporate Research Center, Bethpage, New York, under subcontract to Lockheed Missiles & Space Company, Inc.

Langley Technical Monitor: Huey D. Carden

16. Abstract

DYCAST is a finite element computer program developed at the Grumman Corporate Research Center (CRC) (with partial support from the NASA Langley Research Center) for the nonlinear transient dynamic analysis of structures. As part of the Computational Structural Mechanics (CSM) research program of NASA, the Applied Mechanics Laboratory at the Grumman CRC is developing capabilities in its DYCAST software system for the analysis of aerospace structures made of composite materials. One result of this development effort is that a curved, linear, thin-walled beam element has been incorporated into the DYCAST library of finite elements. The formulation of this new element, called the WARP element, includes both shear deformations and non-uniform torsion (restrained warping).

This report describes the basic concepts, the input data, and several example problems essential to the DYCAST analyst for the successful use of the WARP element.

17. Key Words (Suggested by Author(s)) Dynamic Analysis, Crash Analysis, Nonlinear Analysis, Large Displacement Analysis, Finite Element Analysis	18. Distribution Statement Unclassified - Unlimited. Subject category 64

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of pages	22. Price
Unclassified	Unclassified		

NASA FORM 1626 OCT 86

For sale by the National Technical Information Service, Springfield, Virginia 22161-2171

NASA-Langley, 1991